

STRESS ANALYSIS USING THE FINITE ELEMENT METHOD

This is a description of some practical issues related to the use of finite element programs to calculate the stress distribution in solid bodies. Earlier we described the six major steps involved in using the finite element method:

1. Establish Governing Equations and Boundary Conditions
2. Divide Solution Domain into Elements
3. Determine Element Equations
4. Assemble Global Equations
5. Solve the System of Equations
6. Verify the Results

Steps 1 and 3-5 were the most time consuming when we tried a direct approach on a very simple problem. Step 2, dividing the domain into finite regions, and step 6, verifying the results, were relatively trivial. In the practical application of the finite element method using commercial software packages, you will usually find that the opposite is true. The most time consuming aspect of solving a problem using FE techniques is usually the task of developing a suitable finite element mesh (meshing). The formulation of the element behavior from step 1 and the formation of the element and global stiffness matrices are performed automatically by the finite element software package. The past ten years has seen great advancements in automated meshing algorithms so that much of step 2 can be performed with just a few clicks of a mouse and a few keystrokes. So what is there left for the engineer to do? The engineer still must formulate the problem and interpret the results, he just doesn't have to spend a lot of time performing the laborious repetitive aspects of the analysis.

Step 1

In order to solve a problem, the first step is to define the geometry of the structure to be analyzed and establish the appropriate boundary conditions. The full three-dimensional geometry can be defined using the solid modeling capability of a CAD package. Many problems are essentially two-dimensional and cross-sections or surfaces of the solid model can be used as the basis for developing a 2D mesh of the geometry. If the problem is truly 3D, then a 3D mesh can be generated from the solid model. The engineer must select an appropriate element formulation to model the structure.

Commercial finite element packages have libraries of different elements that can be used to model the structure. The elements can be 1, 2 or 3-dimensional. The underlying behavior of the element can be linear, parabolic or higher order. The order of the element determines how the dependent variables defined at the nodes of the elements vary across the element. For example, linear elements used in stress analysis are formulated to allow the displacements of the nodes to vary linearly along the sides of the element. The strains are related to the derivative of the displacements; therefore, linear elements are sometimes called constant strain elements since the derivative of a linear displacement function is a constant. A parabolic element has a quadratic displacement function and a linear strain distribution.

In order to model a quadratic displacement function, a parabolic element must have three nodes per side whereas a linear element only has two nodes per side. A single, 2D parabolic element will usually do a better job of modeling the displacements over a region than four linear elements. Consider the situation in Figure 1 where a parabolic element is used to model a region. In this element, the displacements at the nodes along one side of the element vary quadratically and the resulting strain field is linear over the region. Using the same nodes plus an additional central node, we can also model the same region with four linear elements. The displacement variation along each side of the element is linear and the strain is constant within each element. This results in a strain distribution over the region that is composed of step functions which are not as smooth as linear functions. The region would have to be modeled with several more linear elements before the strain distribution would become approximately linear. You can see how this becomes important when trying to approximate the stress distribution in an area where there is a steep gradient such as at a stress concentration. Many more linear elements would be required to accurately capture the stress distribution than higher order elements. No matter which order of element is chosen; however, a sufficient number must be employed to capture the underlying stress distribution.

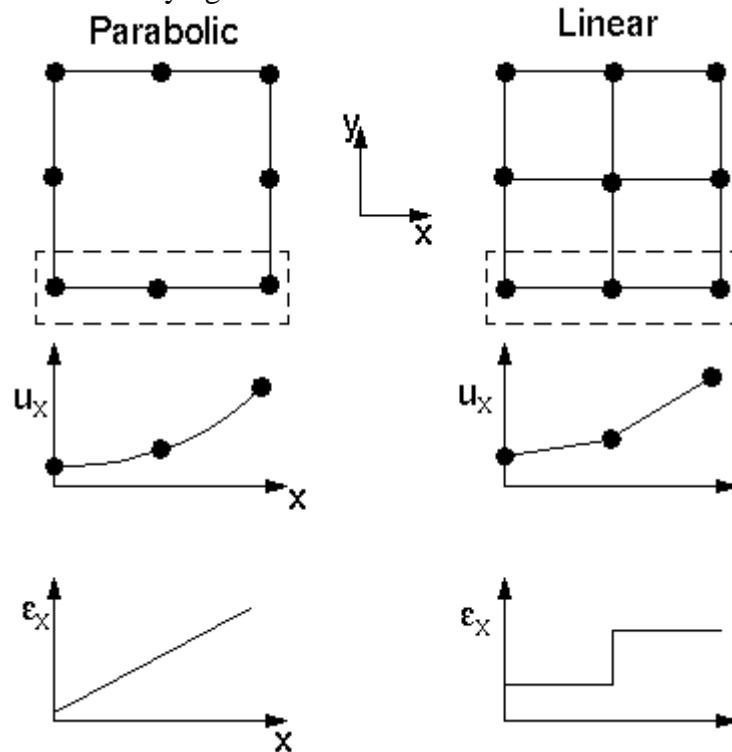


Figure 1 A comparison of the displacements and strains produced in parabolic and linear elements.

Elements have been formulated to model behavior from simple 1D springs to 2D plates, shells and beams. Beam elements are very efficient for modeling a structure composed of beams joined together. For example, it is not necessary to model the detailed geometry of the web and flange with 2D plate elements as in Figure 2. A beam

element permits you to specify the cross-sectional dimensions of the beam and the element itself is formulated to incorporate the axial, torsional and flexural behavior of a beam. Similarly, shell elements are formulated to capture the in-plane stiffness as well as the out-of-plane bending stiffness of thin structures.

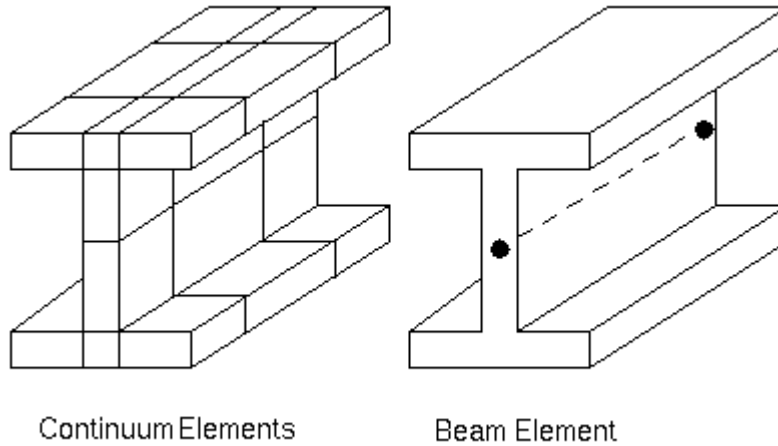


Figure 2 A segment of a beam can be more efficiently modeled as a single beam element as opposed to many continuum elements.

Step 2

Once you have decided upon appropriate elements for your model, you must instruct the program how to divide the region into elements. Most packages now support free meshing routines where you simply specify the region or surface to be meshed and the desired size of the elements. The program then automatically breaks the region into elements of the specified size. In most cases you will want to exercise a little more control over the size of the elements in a particular region. You have the option of specifying local element sizes in selected areas of your structure such as around discontinuities that give rise to stress concentrations.

Many structures that need to be analyzed have symmetry planes. For example, the stress distribution for a simply supported beam with a uniformly distributed load is symmetric about a plane perpendicular to the axis of the beam. The solution on one side of the center plane is the same as that on the other side. Instead of modeling the entire beam, you can model one half of the beam and enforce appropriate boundary conditions on the symmetry plane to reflect the symmetric loading.

In many practical problems, the determining the boundary conditions represents the most difficult part of the problem. Quite often the exact forces that are applied to a structure are not well quantified or understood. In this case, the engineer must make educated guesses about the loading and then verify the results of the analysis by comparing his predictions with experimental measurements made on a prototype of the structure.

A common mistake in finite element analysis is to under-constrain or over-constrain the structure. Every structure must be properly constrained in space so that the reaction forces that develop at the constraints can balance the applied forces. Otherwise,

the structure will not be in static equilibrium and can go flying off into space. Many programs will give an error or warning message when this occurs but some may not. If an analysis completes but you don't have any reaction forces or stresses in the structure when you look at the analysis results, check your boundary conditions. Make sure the model is properly constrained and that forces or displacements are prescribed at the appropriate nodes.

Steps 3-5

Once the model has been meshed and appropriate boundary conditions have been applied, you instruct the program to solve the problem. The software will then automatically take care of forming all of the element stiffness matrices and assembling the global stiffness matrix. Usually there will be some output to describe this process as it is taking place. This gives you an appreciation for the total number of algebraic equations that are being assembled and the number of variables (degrees of freedom) to be solved. It is also during this phase that warning and error messages are generated that might indicate a problem with how you have put together your model. Make sure you ask the program to report any errors and warnings. Oftentimes a program will report errors along the way but still calculate some results for you anyway. Don't just blindly assume that the results are correct - they probably aren't. You should fully investigate the error messages and eliminate the problems that caused them in the first place. (As you probably know by now, a computer is one of the most efficient methods for generating reams of wrong answers. Both good engineers and bad engineers generate errors using the computer. The difference is that the good engineer learns how to separate the good answers from the bad.)

Step 6

A finite element analysis can produce enormous data files that contain all of the results from the analysis. For every node, the results can include:

- displacements
- rotations
- reaction forces
- temperature (if a thermal analysis is performed)
- other field variables.

For every element, the results include:

- the Cartesian stress components,
- the principal stress components
 - the maximum shear stress, the von Mises stress
- the Cartesian strain components
- the principal strain components
- maximum shear strains
- energy components
- other parameters, depending upon the analysis performed.

Postprocessing is the word used to describe the analysis of the results data. Fortunately, it is rarely necessary to wade through table upon table of data. Most commercial software packages have special capabilities for presenting the results in a wide array of formats from vector plots to deformed mesh plots and contour maps. The results can be examined interactively on the computer screen. Along with meshing, postprocessing consumes the lion's share of the effort involved in a finite element analysis. As an engineer, you must decide whether the answers make sense based on the behavior you expected. Here are a few simple questions you should answer when examining the results from an analysis:

- Do the reaction forces balance the applied forces?
- Do the reactions act in the expected direction?
- Does the model deform the way you intended based upon the boundary conditions?
- How do the results compare with back-of-the-envelope calculations from strength of materials?

These are just a few of the questions that must be considered when looking at the results of an analysis. Once you are satisfied that the results seem reasonable, you should refine the mesh to see if the results have converged to an acceptable level. You may have to refine the mesh in areas of discontinuities to properly model the stress gradients that occur around discontinuities.

The finite element method is a very powerful numerical technique for approximating the solution to the governing differential equations of solid mechanics, thermal sciences and other engineering applications. The same basic steps apply regardless of the specific type of analysis being performed. Remember, the final result is only as good as the input provided by the analyst. It is the analysts responsibility to ensure that the results make sense and solve the original problem correctly.